Validation of Reinforced Concrete Modeling Capabilities for Seismic Response

Steven W. Alves
(925) 423-2391
alves6@llnl.gov

This project assesses the ability of our explicit solid mechanics codes DYNA3D/ParaDyn to simulate the seismic response of reinforced concrete structures. These codes were originally created to simulate rapid, large-deformation response. For example, the concrete model was created to assess shock loadings on reinforced concrete structures. In contrast to shock loading, structures subjected to seismic loading will likely go through multiple load reversals over a longer response period.

Recent shake table tests at the University of California, San Diego (UCSD), provide a significant assessment case. A full-scale seven-story slice of a reinforced concrete building was constructed on the Large High-Performance Outdoor Shake Table (LHPOST) at UCSD (Fig. 1). This facility is part of the Network for Earthquake Engineering Simulation. The structure was subjected to four earthquake ground motions of successively increasing amplitude.

Project Goals

Previous work has shown that there are difficulties in modeling seismic response of a reinforced concrete structure using the homogenized concrete/rebar model implemented in DYNA3D/ParaDyn. An instability in the model implementation leads to non-physical stress states and subsequent inversion of elements that terminates the simulation. This misbehavior appears to be more likely with the long duration and load reversals necessary for a seismic simulation.

The goals of this project were to improve the material model and then continue the assessment of the model by comparison with the shake table experiments. The latter included refinement of the boundary conditions and material properties in the finite element model to obtain a better representation of the structure.

Relevance to LLNL Mission

Simulation of the seismic response of reinforced concrete structures is of importance to LLNL's Nuclear Fuel Cycle & Reactor Program, the Global Nuclear Energy Partnership (GNEP), and the National Ignition Facility (NIF). In general, validating this capability would allow LLNL to attract new projects in the area of seismic analysis. This project also promotes collaboration with UCSD.

FY2007 Accomplishments and Results

The numerical robustness of the concrete model in DYNA3D was investigated. The code failure was determined to occur during calculation of a quantity related to the effective plastic strain increment. This error condition can be eliminated by scaling the stress in an element to zero. This improvement was implemented, but it should be noted that large effective plastic strain increments can still be calculated for an element causing large deformations. While this does not prevent simulations from running, the user must assess whether the computed response is significantly affected before using the results. Other corrections to the material model, which were identified during the investigation of the instability, were also implemented.

A finite element model of the structure was generated with separate concrete properties for each level of the building. The properties were based on concrete test data provided by UCSD. Also, rotational stiffness of the shake table platen was included in the model.

Figure 1. Seven-story structure on shake table.

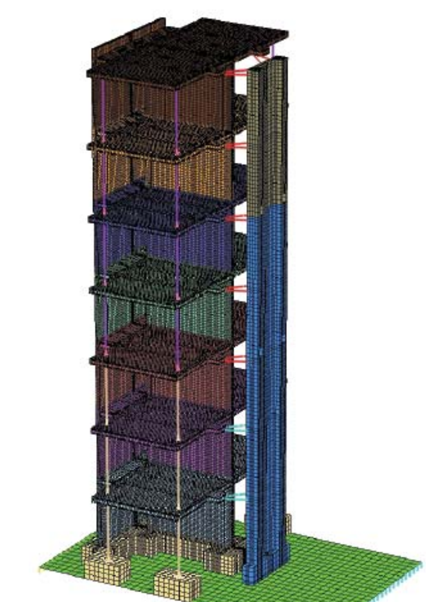

Figure 2. Finite element model of the structure and shake table.

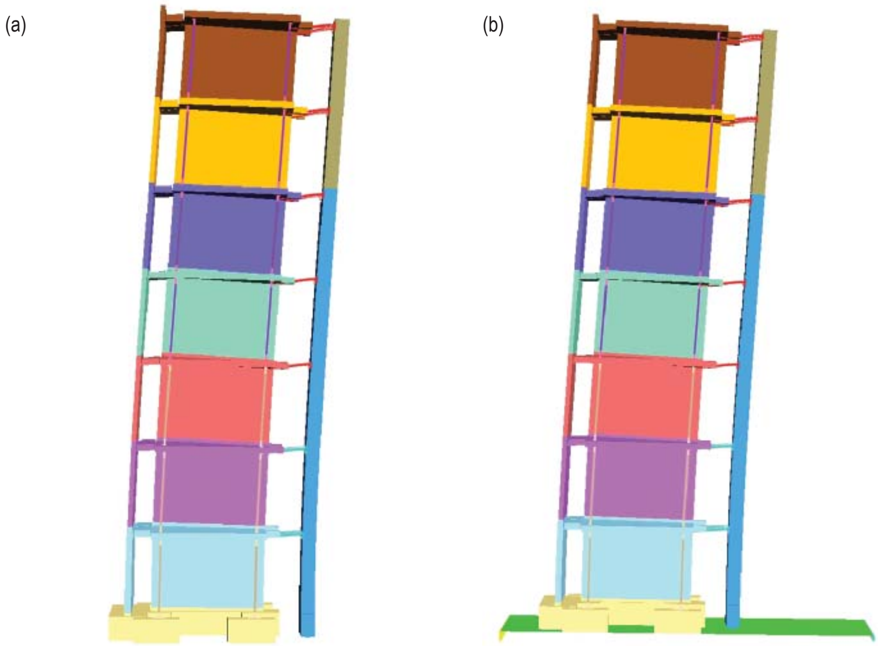

Figure 3. Longitudinal mode at 2.51 Hz with no table (a) and at 1.97 Hz with table rotational stiffness (b).

Both of these features are believed to be important factors in modeling the seismic response of the structure on the shake table.

A linear version of the finite element model was created for running static analyses and then determining the natural frequencies and mode shapes using NIKE3D (Fig. 2). These analyses illustrate the importance of including the rotational stiffness of the table platen in the model. The first longitudinal mode of the structure ranges from 2.51 Hz with no table rotation to 1.97 Hz with a relatively low rotational stiffness of the table (Fig. 3).

Dynamic simulations were performed using ParaDyn for a finite element model with concrete and rebar, using the nonlinear material models. The stiffness of the structure is not captured during the first earthquake motion (Fig. 4) unless the rotational stiffness of the table platen is reduced to a level that is a lower bound for the rotational stiffnesses estimated by UCSD from recorded motions. While the agreement of the modeled response with the experiment is good for the first earthquake, the simulation deviates for the subsequent earthquakes by under-predicting the response (Fig. 4). A possible explanation for this is that as the structure is progressively damaged and the stiffness decreases, the prescribed damping becomes too high in the model.

It is believed that these results partially validate the use of the homogenized concrete/rebar model in DYNA3D/ParaDyn for seismic simulations of reinforced concrete structures. While the numerical instability causing code termination has been corrected, there is still concern about the large deformations caused by the remaining numerical anomaly. The approach is believed to be viable for simulations to determine moderate levels of deformation and damage, but performing simulations for long durations after sustaining significant deformation and damage levels is of questionable value. Obviously, attention must be paid to accurately defining boundary conditions, material properties and damping. Perhaps damping should be modified as the structure is progressively damaged.

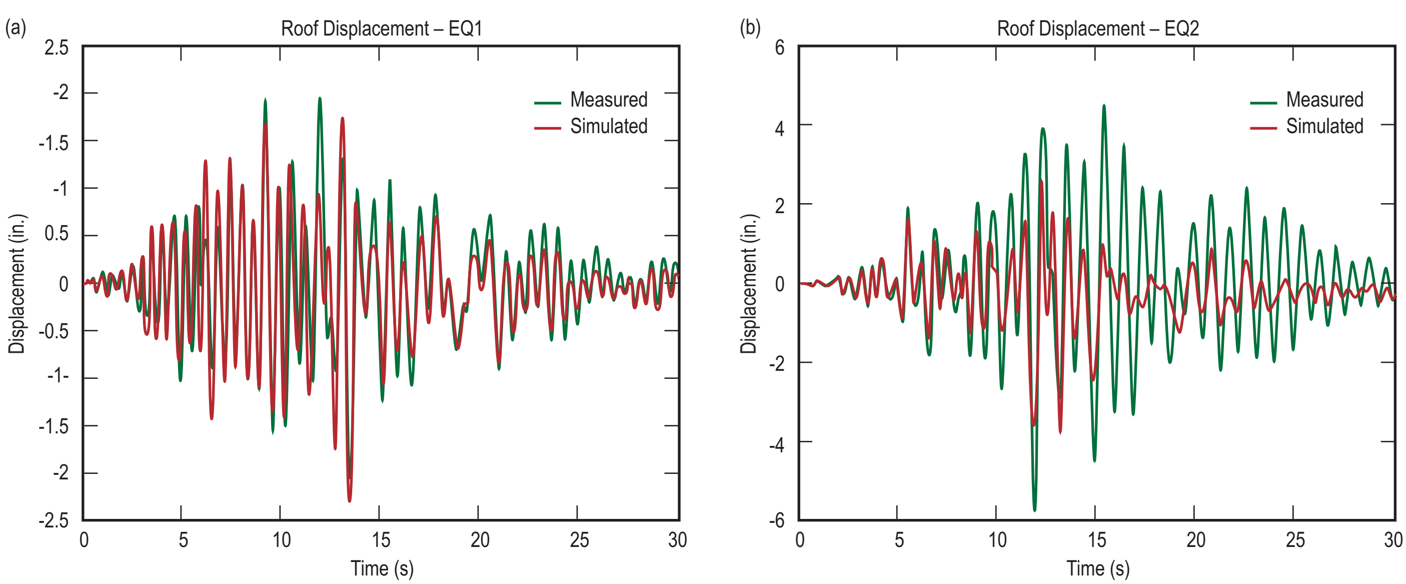

Figure 4. Comparison of simulated roof displacement to measured roof displacement for the first (a) and second (b) earthquakes. (Note difference in vertical scales.)